# **SOLIDWORKS PLASTICS**

### Introduction



SolidWorks Plastics is an add-on feature to the standard SolidWorks package. It is a computer-aided engineering (CAE) program used to calculate and run injection molding simulations, allowing for examination of flow patterns, weld lines, and cooling rates. This assignment aims to introduce the basic functionality of the program and to allow you to explore the ideas behind design for injection molding.

### Assignment

Following the walkthrough below, create an injection molding simulation using SolidWorks Plastics. You are encouraged to use the part you will be injection molding. Please submit a PDF with pictures of some of your plots and describe any interesting things you try or find.

#### Walkthrough

1) Design a part as you would normally model in **SolidWorks** with features like extrude, revolve, loft, etc (get as fancy as you would like). For this demo, we will download the sample part from MEAM.Design.

<b>∂S SOLID</b> WORKS	File Edit Vie	w Insert	Tools	Simulation PhotoView 360	Window	Help 🧟	🗋 • 🔌 •
Extruded Revolved Boss/Base Boss/Bas		ie Extr	5	SOLIDWORKS Applications (press Products SOLIDWORKS Plastics Toolbox		Pattern	🗄 Rib 📓 🚰 Draft 😡 🗎 Shell 🗐
Features Sketch	Evaluate DimXper	t Render	5	folidCAM Row Simulation	, 1	IDWORKS N	IBD Flow Si

- 2) Check to make sure **SolidWorks Plastics** is turned on.
  - a. Select **Tools** from the action bar (pictured above)
  - b. Select Add-Ins... in the dropdown
  - c. Check the boxes for SolidWorks Plastics in the window
    →
  - d. Press OK



- 3) Select the **SolidWorks Plastics** tab on the ribbon.
- 4) Select Getting Started Wizard and a side panel will appear to the right of the screen.



- 5) Select Next.
- 6) You will be prompted with the question of which type of mesh you would like to make. In our case, we will select **Shell Mesh** because of the thin walls in the part.
- 7) The manager panel (left panel on SW) will now prompt us in making the mesh.
  - a. Ensure that the Export Solid Data for Warp Analysis box is checked
  - b. Click the blue arrow
  - c. Domain tab will appear
  - d. The part should be listed in the window labeled as a cavity.
  - e. You can show and hide the cavities to ensure they are connected
    - i. If the part is broken up into multiple parts, go back to your model and check that each feature of the part is merged.
  - f. Click the **blue arrow**
  - g. Surface Mesh tab will appear
  - h. Adjust the **Triangle size** and select **Mesh**. You will want to iterate through several different triangle sizes to find one the covers the part in a tight mesh similar to the middle example below.







- i. Click the **blue arrow**
- j. **Summary** window will appear providing feedback on the mesh. For a good part the aim would be to keep intersections and non-manifolds at zero. →
- k. Click the green check
- I. Click the green check again
- 8) Move back to the **Wizard** panel (right panel in SW) and select **Next**.
- 9) The Material tab will open.
  - a. Select Change Material
  - b. The **Polymer** window will appear (below). You will be able to select from a large range of plastics for your simulation. In this demo, we will use **PS** (polystyrene).
  - c. Click OK in the Polymer window
  - d. Select Next on the Wizard panel (right panel in SW)



- 10) The Process Parameters tab will open.
  - a. Select Change Fill Settings
  - b. The **Fill Settings** Panel (left panel in SW) will appear. You have the control to change the temperature and pressures of the injection.
  - c. Set **Injection Pressure Limit** to 80 MPa to match the Morgan Press G-100T.
  - d. Click the green check in the Fill Settings panel
  - e. Select Next on the wizard panel
- 11) The Injection Location tab will open
  - a. Select Add Injection Location



\$		ß	\$	۲	(9)
Sh	ell Mesi	h - Man	ual		(?
Summ	ary				
W	/aterpro	of		Yes	
N	umber o	f Mesh	Group	1	
E	ement Ir	ntersect	ions	0	
N	on-mani	ifold		0	
В	ad Eleme	ents		0.07	7%
v	ery Bad I	Element	s	0.00	0%
N	laximum	Aspect	Ratio	15.6	53
	nmatche	d Flam	ents	0	

- b. The **Injection Location** panel (below) will appear. This is where we set the injection spot/s for the part.
- c. Click the area on the part where you would like the injection point.
- d. Select the Add Location button (needle icon) on the Injection Location Panel
  - i. It is possible and often useful to add multiple injection points. We will only be using one, however.
- e. Click the green check in the Injection Location panel.
- f. Select Next on the wizard panel



12) The Run tab will open.

- a. Select Run Simulation
- b. Watch as the injection calculates and the part fills.
  - i. Note: this may be SLOW...

Model		Analysis Program	Execution State	
Default		SHELL - FLOW/PACK	Running	
Suspend Resume Help		Show Log Display Partial Results Close after finished Always on top		
		14732		
No. of Elements No. of Part Eleme No. of Sprue/Rur Thickness Factor Coordinate Factor.	nts ner Elen	nents = 0 = 1.00		
No. of Part Eleme No. of Sprue/Rur Thickness Factor Coordinate Factor.	nts her Elen /2016 a	nents= 0 • 1.00 = 1.00 malysis is beginning		

13) Once the analysis is complete..

- a. Click the **play** button on the **Results** panel (Left Panel in SW) and watch the fill simulation from multiple angles and look at the different feature.
- b. The **Results Advisor Window** (left) also provide feedback on the part and some explanation to help read your results.
- c. Select Some of the different plots from the list
  - i. The **Fill Time** plot maps the time as the plastics flows through the part.
  - ii. The **Cooling Time** plot shows time it take sections of the part to cool. In our case we may be concern with **warping** in the lettering.
  - iii. The **Sink Marks** plot shows the measurable effects of the warping that could occur.
  - iv. The **Gate Filling Contribution** plot maps the location of material for multiple injection points.
  - v. The **Ease of Fill** plot is the check off if the part is good to go.
- d. Checking the **Weld Lines** box shows where two flows of material met. These are key weak spots in the part that may require redesign.
- 14) Now play around and try different things like...
  - a. Move or add injection sites
  - b. Change the materials
  - c. Edit some features on the part
  - d. Make an assembly with a flow channel and fill multiple parts at the same time
  - e. Run the simulation on your own part







## **Good Luck!**

Adviser:	
This part can be successfully filled with an	
injection pressure of 11.6 MPa (1677.06 psi).	
The injection pressure required to fill is less	10
than 66% of the maximum injection pressure limit specified for this analysis, which means	
you are well under your specified limit.	
You may be able to reduce the part thickness	
and decrease cooling time, but be sure to	
run an additional analysis after changing the	-



5